Laminar, Two-dimensional Couette Flow

1 Introduction

This case provides a description for two-dimensional Couette flow with constant properties, and a zero pressure gradient.

2 Domain

The two-dimensional geometry for this tutorial is captured in Figure 1 where the rectangular domain is defined by the height, H, and length, L. The streamwise and vertical velocity are defined as u_x and u_y, respectively.

The top surface is a no-slip wall boundary specification $u_x = u_b$, where u_b is a bulk velocity of the top moving wall and $u_y = 0$. The bottom surface is also a no-slip wall boundary specifications with $u_x = u_y = 0$. Finally, the left and right surfaces are periodic. In absence of any external body forces, the flow is aligned to the x-axis and is strictly a function of the vertical-dimension, y, i.e., $u_x = f(y)$.

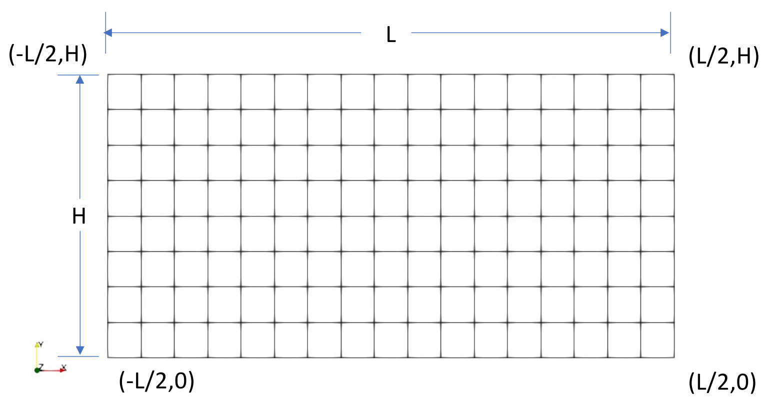


Figure 1: Two-dimensional couette flow in which the height is 2 m and length, 1 m

3 Theory

The variable-density low-Mach equation set is defined by the continuity and momentum equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0. \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j} - \frac{\partial \sigma_{ij}}{\partial x_j} = 0. \tag{2}$$

In the above equation, ρ is the fluid density and u_j is the fluid velocity. The stress tensor is provided by

$$\sigma_{ij} = 2\mu S^*_{ij} - P\delta_{ij}, \tag{3}$$

where the traceless rate-of-strain tensor is defined as

$$S^*_{ij} = S_{ij} - \frac{1}{3}\delta_{ij}S_{kk} = S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}.$$

In a low-Mach flow, the above pressure, P, is the perturbation about the thermodynamic pressure, P^{th}.

3.1 Analytical Velocity Profile

Given the assumptions provided in the introduction, the streamwise velocity equation reduces to,

$$\mu \frac{d^2 u_x}{dy^2} = 0. \tag{4}$$

We note that a more interesting Couette flow can be derived in the presence of a constant pressure gradient. Activation of a non-zero dynamic viscosity, Equation 4 can be integrated twice to obtain,

$$u_x(y) = k_1 y + k_2, \tag{5}$$

where k_1 and k_2 are constants of integration that are obtained through the application of boundary conditions, $u_x(y = 0) = 0$ and $u_x(y = H) = u_b$. Therefore, the final expression for the streamwise velocity is simply a linear function of vertical distance,

$$u_x(y) = \frac{u_b}{H} y. \tag{6}$$

4 Results

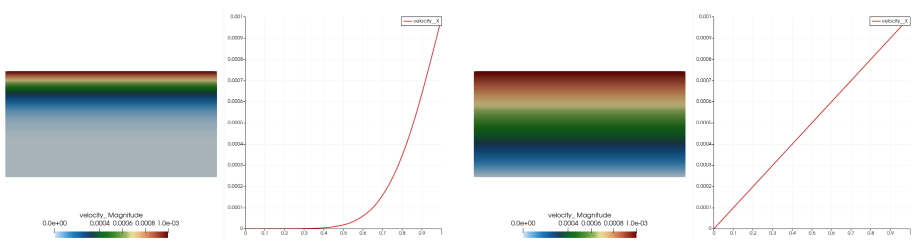
Let us test a simulation in which a the Reynolds number based on height of the domain, assuming the properties of water (ρ =1000 kg/m^3 and μ =8.9e-4 $Pa - s$) at a bulk top velocity of u_b =1e-3 m/s, is approximately 1236.

$$Re = \frac{\rho u^b H}{\mu}. \tag{7}$$

4.1 Simulation Specification and Results

The mesh exercised activates a Quad9 topology, thereby exercising a quadratic underlying basis that yields a nominal third-order spatial accurate simulation.

In Figure 2, results are provided for the specifications provided above.



(a) (b)

Figure 2: Velocity shadings (left) and velocity profile (right) for the $Re = 1236$ case.

5 Discussion Points

There are several interesting activities associated with this sample case including the following:

- Ensure that derivation of Equation 6 is clear.
- Explore the mesh and input file specifications associated with this case.
- In Figure 2, the flow results demonstrate a linear profile, as expected. Based on past experience with a linear basis, comment on the usage of a quadratic basis.
- Probe all degree-of-freedom results, i.e., velocity and pressure. What is of interest?